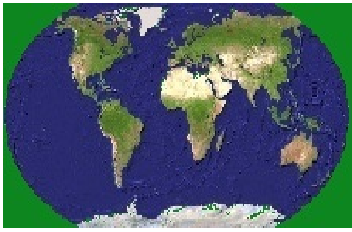

International Journal of
Multidisciplinary Research and
Development

IJMRD 2014; 1(5): 1-4
www.allsubjectjournal.com
Received: 12-09-2014
Accepted: 24-09-2014
e-ISSN: 2349-4182
p-ISSN: 2349-5979

M.Y. Abdollahzadeh Jamalabadi
Assistant Professor, Maritime University of Chabahar, Chabahar, mail. Box: 99717-56499, Iran Senior Researcher, Graduate School of Mechanical and Aerospace Engineering Gyeongsang National University, Jinju, South Korea

Correspondence:
M.Y. Abdollahzadeh Jamalabadi
Assistant Professor, Maritime University of Chabahar, Chabahar, mail. Box: 99717-56499, Iran Senior Researcher, Graduate School of Mechanical and Aerospace Engineering Gyeongsang National University, Jinju, South Korea.

Effect of fuel inject angle on non-premixed combustion of air/methane mixtures in vertical cylinder

M.Y. Abdollahzadeh Jamalabadi

Abstract
The present work is devoted to the study of non-premixed combustion within a vertical cylinder using a turbulence model. A study of the effect of inlet gas angle on methane/air combustion is performed. The results show that an increase in angle of inlet gas enhances the mixing rate, peak temperature, and water and carbon dioxide volume fraction inside the middle region of the chamber. The locations of the maximum temperature and product concentration shift closer to the combustor inlet with an increase in inlet angle.

Keywords: Methane/air Flame, Turbulent model, Numerical simulations, Non-premixed, turbulent combustion.

1. Introduction

In coal furnaces, diesel internal-combustion engines and pool fires there is non-premixed combustion, in which there are two distinct streams of fuel and oxidizer before burning [1]. Generally the non-premixed combustion flames contain locally fuel-rich regions where because of lack of oxidizer is the all set for the pollutant productions. At those regions products yield larger hydrocarbons instead of being to carbon dioxide and water [2-3]. The soot formation is sturdily hooked on in the fuel concentration and temperature [4-5].

In some combustion chambers in laminar and simple turbulent jet diffusion flames the injectors located in the manifold and the fuel are injected along with the angle of the injector and the spray characteristics of the injector is important to obtain the optimum combustion performances. The results of numerical modeling of direct axisymmetric turbulent methane-air simple jet (without swirl) flames are present in many numerical studies which normally have shown good agreement with the experimental measurements [6]. Effect of fuel flow direction is useful in many technical applications, particularly in furnaces and gas turbines to improve flame stabilization, ignition stability, mixing enhancement, pollutant reduction and blow-off characteristics [7]. Although the effect of many parameters on combustion of air/methane mixtures in has been investigated by many investigators, but few researches are performed the influence of the injection angle strategies on combustion processes have been reported [8-10]. Also the effect of surface thermal radiation effects which is important in other fields are not considered [11-14]. The aim of this paper is to study the effect of fuel injection angle on fluid, mass transfer, and thermal characteristics of a methane/air flame in turbulent diffusion flames at vertical cylindrical combustor.

2. Computational domain

A common fuel injection valve of a combustion chamber, including a concave conical surface, fuel injection holes in its surface, and a portable needle valve. But here that valve is not modeled through the combustion chamber and just its effect on the fuel injection angle is considered. As shown in the Figure 1 a vertical combustor with a circular cross-section is assumed to confine the flame and prevent gas composition fluctuations subsequent from the ambient air. The combustion chamber is 25 cm in diameter (D) and 100 cm in length (L) that a one cm diameter (d) hole at the center is used to deliver the fuel (CH_4) with the velocity magnitude of 80 m/s to the burner and the air ($0.23\ O_2\ 0.77\ N_2$) is entered from the annular between that hole and burner with the velocity of 20 m/s. The fuel injection angle is measured from the axis of symmetry of the combustion chamber. The wall and inlet hold at environment temperature (300 K).

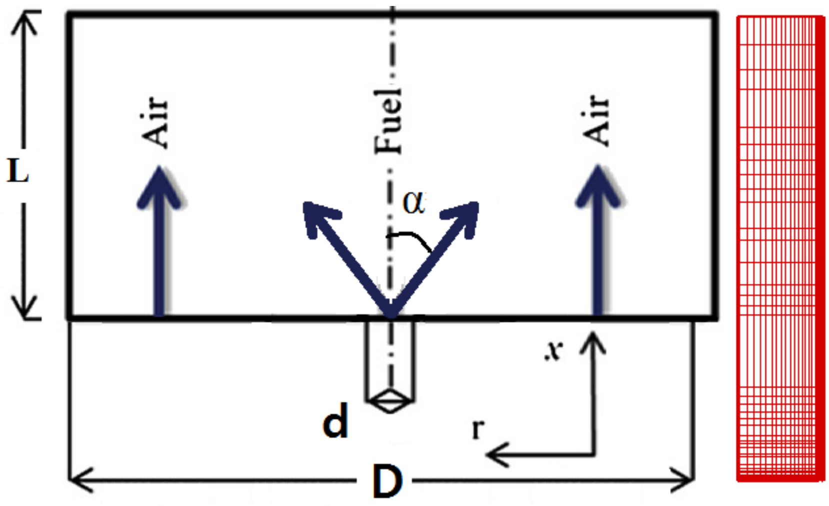

Fig 1: Schematic of the vertical cylinder combustion chamber and the computational mesh.

2.1 Mathematical Modeling

In this study the combustion will be modeled assuming complete conversion of the fuel to CO_2 and H_2O. The mass, momentum, energy, and species conservation equations are where the turbulent stresses are calculated from an algebraic stress model and wall-function approach is used in the near-wall. The computational mesh of the half of the burner with boundary layer meshes the inlet and the axis of symmetry is illustrated in figure 1.

$$\frac{\partial u}{\partial x}+\frac{1}{r}\frac{\partial}{\partial r}(rv)=0 \qquad (1)$$

$$\frac{1}{r}\left[\frac{\partial}{\partial x}(r\rho uu)+\frac{\partial}{\partial r}(r\rho uv)\right]=-\frac{\partial p}{\partial x}+\mu\nabla^2 u-\frac{1}{r}\frac{\partial}{\partial r}(r\rho\,\overline{u'v'})-\frac{\partial}{\partial x}(\rho\,\overline{u'u'}) \qquad (2)$$

$$\frac{1}{r}\left[\frac{\partial}{\partial x}(r\rho uv)+\frac{\partial}{\partial r}(r\rho vv)-\rho w^2\right]=-\frac{\partial p}{\partial r}+\mu(\nabla^2 v+\frac{v}{r^2})-\frac{1}{r}\frac{\partial}{\partial r}(r\rho\,\overline{v'v'})-\frac{\partial}{\partial x}(\rho\,\overline{u'v'})-\frac{1}{r}\rho\,\overline{w'w'} \qquad (3)$$

$$\frac{1}{r}\left[\frac{\partial}{\partial x}(r\rho uh)+\frac{\partial}{\partial x}(r\rho vh)\right]=\Gamma_h\nabla^2 h-\frac{1}{r}\frac{\partial}{\partial r}(r\rho\,\overline{v'h'})-\frac{\partial}{\partial x}(\rho\,\overline{u'h'})+\dot{S}_h \qquad (4)$$

$$\frac{1}{r}\left[\frac{\partial}{\partial x}(r\rho um_j)+\frac{\partial}{\partial r}(r\rho vm_j)\right]=\Gamma_{mj}\nabla^2 m_j-\frac{1}{r}\frac{\partial}{\partial r}(r\rho\,\overline{v'm_j'})-\frac{\partial}{\partial x}(\rho\,\overline{u'm_j'})+R_j \qquad (5)$$

3. Results and Discussion

The conservation equations are solved using the OpenFOAM with the method described in [15]. Figure 2 shows the simulated contours of axial and radial velocities, stream function, water (H_2O), carbon dioxide (CO_2), and nitrogen (N_2) mass concentrations inside the combustor for fuel injection parallel to the axis of the burner. As shown the radial velocities are in the one tenth orders of the axial velocities, and stream function generally is parallel to the axis of the combustion chamber. The mass concentrations of the oxygen (which is not shown here) are similar to the nitrogen and generally divided to the two regions of inlet and chemical reactions. The most of the nitrogen exited from the cavity near the wall and the minimum amount was occurring at the axis of its. The fuel is consumed in the small region near to the inlet and for the products of the reaction (water and carbon dioxide) there are three regions in the chamber which are zero at air inlet and a maximum at the outlet near the wall of the chamber and in moderate concentration in other sides. The results show that an increase in injection angle cause to create the four regions for the water and carbon dioxide in the chamber which are zero at air inlet and a maximum in the middle of the chamber and in moderate concentration in other sides. Because there are not a great change in the pressure inside the chamber that contours are not presented here.

The figure 3 shows the temperature distribution caused by heat release at various fuel injections. Because the in cylinder pressure is mostly constant, that is not shown here. As illustrated the increase of the inject angle increases combustion efficiency, due to enhancement of mixing rates between the fuel and oxidant and augment the combustor peak temperature. However, the variation for the first injection is slightly higher. Furthermore, the increase of the inject angle causes the combustion zone of the diffusion flame enlarged and the location of the maximum temperature shifts inside the chamber. In addition, for the narrower-angle injector, fuel jets are restricted in a smaller region than the wider-angle injectors, and the mixing would be less homogeneous.

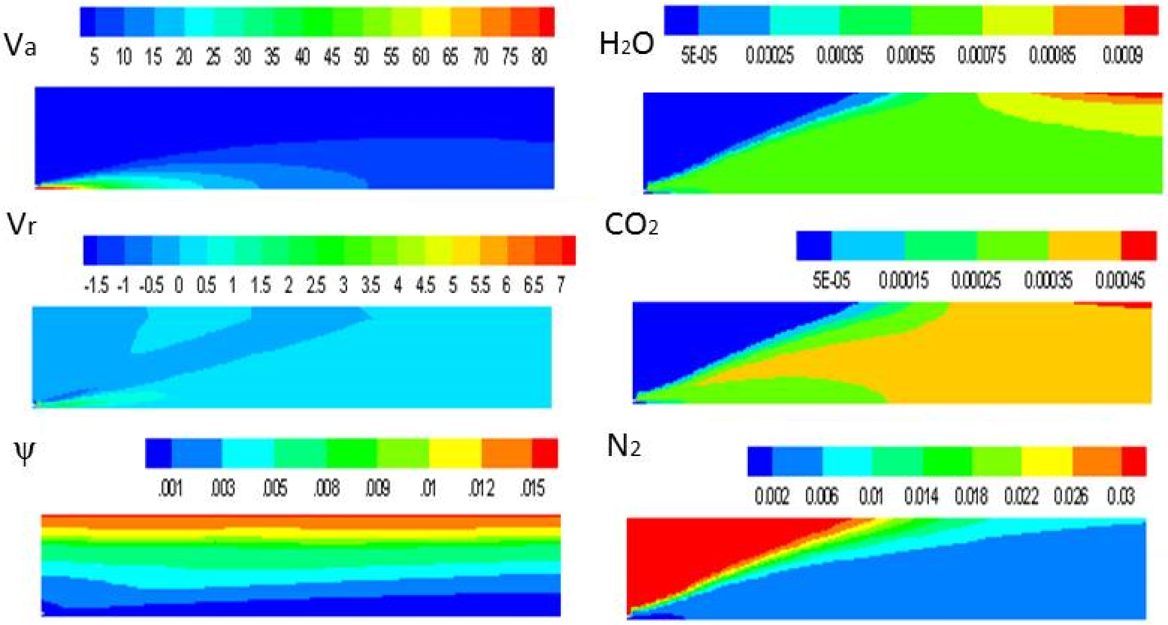

Fig 2: Axial and radial velocities, stream function, H_2O, CO_2, N_2 mass concentrations.

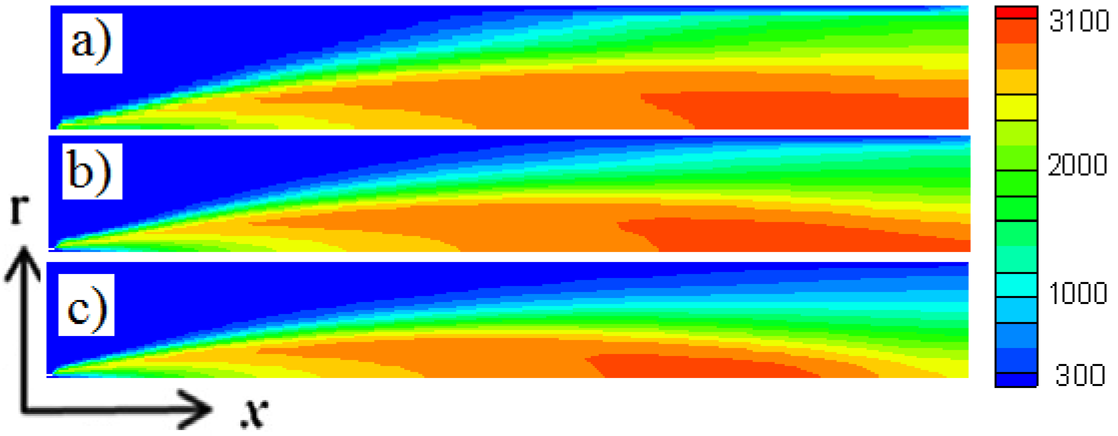

Fig 3: Temperature contours at various fuel injection angle a) α=0 b) α=45° c) α=80°

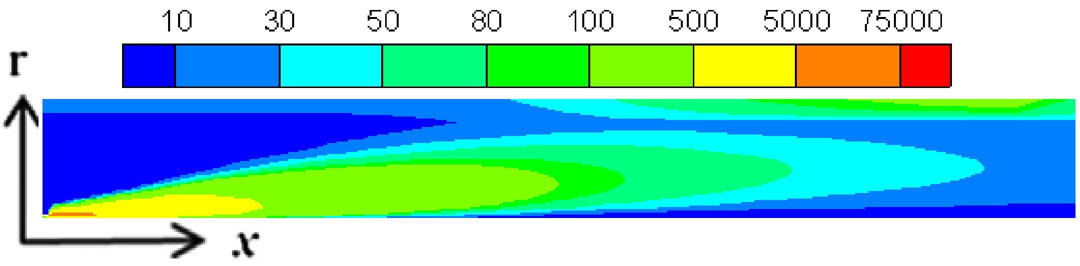

Fig 4: Vorticity contour at α=0

The figure 4 shows the vorticity distribution at fuel injection parallel to the combustion chamber axis. As illustrated the maximum of the vorticity occurs in the methane inlet because of the high mixing of the fuel and oxidizer.

4. Conclusions

Burning behaviors of non-premixed methane-air flame are studied for different fuel injection angles. As shown the fuel injection angle is an efficiency parameter to control the fluid flow and combustion characteristics through the combustion chamber.

5. References

1. Turns SR. An Introduction to Combustion: Concepts and Application, McGraw-Hill Science, 2006.
2. Brookes SJ, Moss JB. Measurements of Soot Production and Thermal Radiation from Confined Turbulent Jet Diffusion Flames of Methane. Journal of Combustion and Flame 1999; 116:49–6.
3. Beltrame A, Porshnev P, Merchan MW, Saveliev A, Fridman A, Kennedy LA *et al.* Soot and NO formation in methane–oxygen enriched diffusion flames. Journal of Combustion and Flame 2001; 124:295–310.
4. Warnatz J, Mass U, Dibble RW. Combustion,Physical

and Chemical Fundamentals, Modeling and Simulation, Experiments, Pollutant Formation. Springer 2006.

5. Zhang J, Nieh S, Zhou L. A new version of algebraic stress model for simulating strongly swirling flows. Journal of Numerical Heat Transfer 1992; 22:49-62.
6. Tiegang F, Robert E, Coverdill E, Chia-fon FL, Robert AW. Effects of injection angles on combustion processes using multiple injection strategies in an HSDI diesel engine. Fuel 2008; 87:3232–3239.
7. Myung YK, Chang SL. Effect of a narrow fuel spray angle and a dual injection configuration on the improvement of exhaust emissions in a HCCI diesel engine. Fuel 2007; 86:2871–2880.
8. Chen SK. Simultaneous reduction of NO_x and particulate emissions by using multiple injections in a small diesel engine. SAE paper 2000-01-3084; 2000.
9. Chui EH, Raithby GD. computation of radiant heat transfer on a nonorthogonal mesh using the finite-volume method. Numerical Heat Transfer, Part B: Fundamentals. An International Journal of Computation and Methodology 1993; 23(3)269-288.
10. Lethner GA, Jacobs TJ, Chryssakis CA, Assanis DN, Siewert RM. Evaluation of a narrow spray cone angle, advanced injection timing strategy to achieve partially premixed compression ignition combustion in a diesel engine. SAE paper 2005-01-0167; 2005.
11. M.Y. Abdollahzadeh Jamalabadi and J.H. Park, Thermal radiation, joule heating, and viscous dissipation effects on mhd forced convection flow with uniform surface temperature, Open Journal of Fluid Dynamics (4) 2 , (2014) 125-132
12. M.Y. Abdollahzadeh Jamalabadi, Experimental investigation of thermal loading of a horizontal thin plate using infrared camera, Journal of King Saud University – Engineering Sciences (26) 2 (2014) 159-167
13. M.Y. Abdollahzadeh Jamalabadi, M.Ghasemi,M.H.Hamedi ,Numerical Investigation of Thermal Radiation Effects on Open Cavity with Discrete Heat Sources, International Journal of Numerical Methods for Heat and Fluid Flow (23) 4 (2013) 649-661
14. M.Y. Abdollahzadeh Jamalabadi, M.Ghasemi,M.H.Hamedi, Two-dimensional simulation of thermal loading with horizontal heat sources, Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 226 (2012) 1302-1308
15. Hassan IK, Khalid MS, Hossam SA, Mohsin MS, Mazlan AW. Implementation of the eddy dissipation model of turbulent non-premixed combustion in Open FOAM. International Communications in Heat and Mass Transfer 2011; 38:363-367.